Prediction of Rotating Stall during Startup for Axial Compressors

Thomas K. Walker
Mitsubishi Heavy Industries America Inc.
20 East Greenway Plaza, Suite 830 Houston, TX 77046, U.S.A.

Masamitsu Okuzono
Mitsubishi Hitachi Power Systems, Ltd.
2-1-1 Shinhama, Arai-cho, Takasago-city Hyogo, Japan

Ryosuke Mito
Mitsubishi Heavy Industries, Ltd. R&D Center
2-1-1 Shinhama, Arai-cho, Takasago-city Hyogo, Japan

ABSTRACT
This paper presents work on using modern computation fluid dynamic (CFD) techniques to predict the number of rotating stall cells during startup and how this number changes for different startup conditions.

Specifically, the CFD showed a different number of stall cells for the normal startup condition (requires one hour at ISO conditions) and the rapid startup condition. In addition, we found that the CFD predicted different stall cell rotating speeds for each condition. Given this, we concluded that modern URANS CFD techniques have some potential in predicting flow behavior at startup and that CFD can be a powerful tool to study under what conditions it is safe to start a gas turbine engine.

INTRODUCTION
Land-based power generation gas turbine engines must be able to startup in very hot and cold conditions and provide a wide range of power at a single shaft speed. In addition, due to the increase in the use of renewable energy, these gas turbines must be able to start very rapidly to provide power when renewables, such as wind power, are not available due to environmental conditions.

Given this, the startup behavior of the gas turbine, in particular the axial compressor where rotating stall occurs during startup, must be well understood in order to reduce the risk to the engine. Rotating stall, in particular the number of stall cells, is of critical importance since coupled with the blade geometry this phenomena determines blade vibration frequencies. As such, poorly predicting the stall cell count can lead to unexpected vibration characteristics and in the worst case catastrophic events like blade resonance.

Decreasing the startup time to increase operating flexibility has several challenges for large industrial gas turbines in addition to the generic issues described above. During startup the temperature and pressure are lower than design, in other words, there is not enough power generated by the turbine to drive the compressor. To counter this, an assist motor is used during startup. As such, to decrease the time needed for startup, a bigger motor or more fuel (to increase temperatures) is required. This will drive up overall costs.

Some possible ways to mitigate this cost are to reduce rotor weight, reduce power required to turn the compressor via an aerodynamic redesign, or to increase the base power of the turbine during startup. Research is being carried out to successfully design a machine with the above qualities, however, weight reduction and aerodynamic improvements are not the focus of this paper.

Even if the above issues of cost are solved, however, there is a worry about the more complex issue of axial compressor stall during startup. Rapid changes in the flow field can lead to varying responses in the metal temperature on each part of the structure, different changes in clearance, and an increase in heat stresses for compressor, combustor, or turbine. In addition, changes in pressure fluctuation, combustion vibration, as well as delayed response of mechanical devices such as the inlet guide vane (IGV), variable stator vanes, or bleed valves may occur.

In summary, the following are the key issues faced during rapid startup for industrial gas turbines.

- Cost issues related to use of bigger motors or more fuel (to increase temperatures)
- Avoidance of severe axial compressor rotating stall
- Prevention of blade rubbing in the compressor or turbine due to an increased rate of thermal expansion and vibration

Although these are all important issues, this paper focuses only on the issue of axial compressor rotating stall during startup as shown highlighted in red in Figure 1.
The axial compressor is designed such that the axial velocity is roughly constant throughout the machine. In addition, the annulus area decreases moving towards the rear of the machine which restricts the mass flow rate during startup due to the need to manage the volume of flow at lower pressure ratios near the choke condition. This causes the axial velocity to decrease in the front stages given the larger flow area. That in turn leads to the development of rotating stall cells due to larger incidences (reduced axial velocity contributes to increased inlet flow angles) compared to the design point.

These stall cells rotate around the annulus in the same direction as the rotor which causes large pressure fluctuations. These fluctuations can lead to large vibrations or even resonance for the blades as well as other surrounding structures. In the worst case, this can destroy the compressor.

Given this, it is clear that rotating stall occurring during startup is undesirable, but for most compressors, it is hard to eliminate completely even for a normal startup cycle. In reality, the goal is to predict and subsequently manage this phenomenon to avoid dangerous vibration frequencies and potentially large pressure fluctuations. These fluctuations can lead to large vibrations or even resonance for the blades as well as other surrounding structures.

In the transient, a rapid startup means a higher pressure ratio due to an increase in fuel or a change in the assist motor configuration. This implies an even larger drop in velocity in the front stages and thus a worsening of the rotating stall compared to the normal startup condition. As such, it is very important to understand exactly how the rotating stall behavior changes for the rapid startup condition and then to devise the appropriate variable vane schedule, blade design, and so on.

For that purpose, in this research we looked to analyze rotating stall behavior during startup using CFD techniques comparing the stalling behaviors for both normal and rapid startup.

BACKGROUND

This research looks to use CFD to evaluate changes in the stalling behavior, such as stall cell count and cell rotating speed for two conditions. If the CFD proves effective, then it may be used in the future instead of much more time consuming and expensive experimental methods to manage this phenomenon during startup. As such, the first step is to evaluate the accuracy of the CFD compared to existing test data in order to clearly elucidate the feasibility of using CFD to perform such a task. Before beginning the calculations; however, we first looked at other examples in literature of using CFD on such a large scale and took any lessons learned and incorporated them into our methodology.

There are many examples in literature of numerical techniques being used to evaluate axial compressor behavior near stall. Focusing on lower fidelity, smaller-scale techniques; the work of Saxer-Felici et al. [1] is of import. That paper, based on previous theoretical and experimental studies, focused on the importance of the fluid inertia and momentum exchange mechanisms to rotating stall. An Euler calculation was conducted on a single stage low speed subsonic axial compressor. This result supported the assumption that the cell structure and general mechanisms of full-span rotating stall are essentially governed by inertial effects and momentum exchange rather than details of turbulence modeling. Advanced techniques for modeling vortex structures, like LES, may not be needed to simulate rotating stall.

In [2], Gan et al. also works with a single stage (NASA stage 35) but employs URANS techniques rather than an Euler code. This simulation indicates that the inception of rotating stall for Stage 35 is spikes. The calculation is able to predict the stall cell growth and stall cell rotating speed. Crevel et al. [3] performed a URANS calculation on a multistage axial compressor then compared that to experimental results. Their study investigated the influence of the upstream and downstream volumes on rotating stall in a high-pressure, high-speed research compressor. The numerical results show that correctly modeling all of the large volumes in the compressor is important to correctly predict rotating stall. Their results showed that properly modeling the compressor led to fewer cells developing in the third stage and the rotating speed being slightly higher compared to more simplified compressor models. The major difference was linked to the presence of large flow volumes causing a one dimensional low frequency oscillation of the static pressure.

Next, several papers which employed advanced turbulence models where examined to help determine the cost benefit ratio for those techniques. Zaki et al. [4], used a hybrid URANS/KES (HRKES) turbulence model on NASA stage 35. Their HRKES model solves Menter’s k-ω-SST equation near the walls and switches to a Kinetic Eddy Simulation (KES) model away from the walls. The KES model solves directly for local turbulent kinetic energy and local turbulent length scales. This alleviated the grid spacing dependency found in other LES/DES models. They found this approach to be satisfactory in terms of predicting rotating stall, but results where only shown for a single stage and the method is much more complex than a URANS calculation.

In [5], Gourdain uses LES not specifically for stall prediction, but more in general to understand any accuracy benefits of LES over URANS. The model was a single stage compressor. They found that LES does have some advantages. Specifically, they found that URANS predicted much higher losses compared to LES as well as the fact that only LES can predict the development of high-energy frequencies in the tip region related to the pulsation of tip leakage flow.
Im et al. [6] studied the use of DES on NASA rotor 67. They reported that rotating stall is initiated by a spike flow disturbance which quickly causes the rotor to stall. This process is captured by full-annulus DES as well as by URANS, but DES predicts the stall inception roughly one rotor revolution earlier. With that particular geometry, DES matches the available data more consistently than URANS.

Finally in Marty et al. [7], Zonal DES (ZDES) was used to compute compressor performance maps and that flow prediction was used to select the geometry for an injection device located upstream of the rotor tip. The flow including the injection device was then simulated using URANS and ZDES. They found that the injection technique reduces the loss area and level, energizing and stabilizing the tip leakage vortex thanks to the introduction of high momentum flow. The details of this flowfield were predicted very well using this ZDES technique.

In summary, it was shown that URANS is capable of simulating rotating stall in a multistage environment. LES/DES as well as other advanced turbulence modeling techniques show some additional advantages especially in the tip region, but have a high cost in terms of calculation time (large meshes are required). As such, these techniques are really more suited for application to single stage or single row calculations as evidenced by the lack of literature using the techniques for large multistage machines.

In summary, it was shown that URANS is capable of simulating rotating stall in a multistage environment. LES/DES as well as other advanced turbulence modeling techniques show some additional advantages especially in the tip region, but have a high cost in terms of calculation time (large meshes are required). As such, these techniques are really more suited for application to single stage or single row calculations as evidenced by the lack of literature using the techniques for large multistage machines.

Given that there is evidence from our previous work at MHI that during startup the stall cells rotate at the hub, and the fact that this research is looking to model the first 8 stages of the compressor, it was decided that the CFD methodology will be URANS carefully modeling all relevant large volumes in order to correctly capture any static pressure oscillations as suggested in [3].

**METHODOLOGY**

In order to evaluate the differences between normal startup and rapid startup, a typical large industrial gas turbine compressor geometry was modeled. Figure 2 shows the entire gas turbine with the compressor boxed in red. Figure 3 shows the schedule for a normal startup. It takes twenty minutes to raise the rotating speed to 100% and about thirty to forty minutes for the power output to reach the desired level.

![Figure 2: Gas turbine compressor under study](image)

The compressor under study has a typical number of stages preceded by an IGV. In addition, the stators in the front stages are variable vanes meaning that the blade angles can be changed during operation to tailor the flow as needed. The variable vanes are always employed during startup as a way to manage rotating stall.

![Figure 3: Gas turbine startup schedule](image)

Moving towards the rear of the machine, there are bleed slots. Each bleed slot’s area can be controlled during operation via a valve attached to the piping system. This valve is opened during startup while the rotating speed is still low and subsequently closed as the speed is increased until the desired bleed flow is reached for normal operation. Controlling the bleed flow is another technique used to manage rotating stall.

The calculation model features a compressor test rig which has an IGV plus eight stages. Based on prior research, which also included four and six stage models, stall was expected to appear somewhere in the first few stages so this model was determined to be sufficient length to keep the exit boundary condition from affecting the stalling behavior. As for pitchwise extent, the model was full annulus in order to remove uncertainty that would be introduced by enforcing periodicity after a certain number of pitches in the tangential direction. Finally, in addition to the compressor gas path and blading, all large volumes were modeled as suggested in [3]. Specifically, that led to the modeling of the intake duct in its entirety. The casing bleed slot as well as the bleed plenum, however, was not modeled because that introduced instability into the calculation. Figure 4 shows the entire calculation model.

![Figure 4: Calculation model’s block arrangement with flow direction indicated](image)
The mesh used for this calculation is known as a smoothed-H type mesh. A meridional view is shown in Figure 5. The CFD code was an in-house multi-block solver meaning that the model was divided such that one block was used for each blade. Several blocks were used to model the inlet duct as is apparent in Figure 4.

In the axial direction, a sliding plane was set in-between each row. There were approximately 400 thousand mesh points in each blade block (axial direction: about 120 points, radial direction: 65 points, pitch direction: 49 mesh points). The entire full annulus mesh including the intake duct was about 380 million cells. The turbulence model was Spalart-Allmaras.

Figure 5: Calculation mesh for the eight stages of the compressor rig

As for the calculation methodology, firstly a RANS calculation was performed at 50% speed. At that rotating speed, the static pressure at the exit boundary of the eighth stage was adjusted until the total pressure matched the test data. Based on detailed measurements, this was a rotating speed and pressure ratio where rotating stall is expected to always occur. Next, an URANS calculation was restarted from the RANS solution at that point and left to run for 10 rotor revolutions. Due to the size and complexity of the model it took a significant amount of calculation time to get the required level of convergence for a given condition. Due to this, our results only consist of two solutions; one for normal startup and one for rapid startup.

CALCULATION RESULTS

Comparison against Experimental Data

Before analyzing the stalling behavior during startup, the accuracy of the URANS calculation was validated against the rig test data for the normal startup condition. Figure 6 shows the compressor inlet mass flow normalized by the design flow vs. the pressure ratio. Experimental data during startup is shown as the black line. First comparing the test data to the RANS calculation, at a similar pressure rise there is about a 10% error in the inlet massflow. Comparing to the URANS calculation, however, the CFD matches the test data operating conditions quite well.

![Figure 6: Operating point during startup vs. test data](image)

URANS proving notably more accurate was expected given the high level of unsteadiness present during startup. Additionally, Figure 7 shows test data measured on the casing and the URANS pressure distribution on the casing after 10 revolutions as the solid line. The URANS matches the data very well whereas the RANS calculation (not shown) had some error owing to the differences in mass flow seen in Figure 6. This confirmed that the compressor stage matching can be captured with sufficient accuracy by the URANS calculation.

Normal Startup

The average axial velocity predicted by the RANS calculation in the meridional plane is shown in Figure 8. There are two areas of low momentum flow that could serve to initiate rotating stall. The first is in the tip of the 1st rotor and is thought to be due to the fact that the IGV is severely closed at this condition. The second is at the hub of the 2nd stator continuing into the inlet of the 3rd rotor.

Figure 8: RANS flow solution on the meridional plane

![Figure 7: Wall static pressure during start up vs. test data](image)
After two revolutions, the flow in the 1st rotor mostly remains periodic while the flow in the 3rd rotor shows signs of stall cell development in the hub. The likely reason why stall initiates in the 3rd rotor rather than in the 1st rotor is because the flow in the 1st rotor simply forms a ring stall pattern around the annulus due to low mass flow caused by the closed IGV. Further downstream in the third stage, however, the blade wakes from all the upstream stages help to increase the overall asymmetry.

Figure 11: Normal startup URANS calculation after 10 rotor revolutions

In addition, local geometry or flow phenomena such as the 3rd rotor hub inlet flow angle or endwall behavior play a part in why the stall develops in the 3rd rotor.

After 10 revolutions, the rotating stall pattern stabilizes and four stall cells are observed. The axial extent of the cells, shown in Figure 11, is from the 2nd stator to the inlet of the 6th rotor. By the exit of the domain after the 8th stage the flow has mostly returned to uniform in the pitchwise direction confirming that using 8 stages was sufficient to model the phenomena seen at the normal startup condition.

Figure 12 shows that the cell structure is symmetrical around the annulus and that the spanwise extent of the low momentum flow in the worst case is greater than 50% at the exit of the 2nd rotor reducing to about 30% after the 3rd rotor.

Although the detailed data itself cannot be published, two high response pressure sensors where placed at fixed locations around the annulus at the exit of the 3rd rotor. From the change in pressure associated with a stall cell, the number of cells as well as the cell rotating speed was determined experimentally. Like the CFD, the test data also showed four rotating stall cells in the hub. The spanwise location of the stall cells was confirmed in recent experiments using high response pressure sensors embedded in the stator blades near the hub. The CFD also matched the stall cell rotating speed seen in the data to within 10% of rotor speed.

Rapid Startup

The rapid startup condition was modeled by first analyzing the available data. The test data showed that there was an increase in pressure ratio compared to the normal startup condition. As such, the rapid startup condition was modeled by restarting the normal startup calculation after the four cell rotating stall pattern had stabilized after the 10th cycle; then, while the calculation was running, the back pressure was slowly increased noting any changes in the flowfield. From the data, we know that the cell count tends to decrease from four cells to one cell for the rapid startup case. Therefore, the question was whether or not the stall cells seen in the calculation would change appropriately as the pressure was increased.

Figure 13 shows the rapid startup calculation after 2 revolutions. The flow field is still mostly unchanged in stages two to five. Since the pressure is being increased at the back of the machine, it was expected that it might take several revolutions before the front stages show any appreciable changes in flow pattern.
Figure 13: Rapid startup URANS calculation after 2 rotor revolutions

Figure 14 shows the solution after four revolutions. The most noticeable change is that the axial extent of the stall cells has extended towards the downstream. Also, the width of the cell in terms of blade pitches has increased.

In short, as the pressure increases, the velocity decreases which at this condition contributes more low momentum flow to the stall cells. After 6 revolutions, the results of which are shown in Figure 15, the axial extent of the stall cells has increased to the point of interfacing with the exit domain and the overall stall cell count has reduced to three. Unlike in the normal startup case, these stall cells are asymmetric around the annulus.

After the 6th revolution, due to an increasing amount of reversed flow interfacing with the exit boundary condition the calculation was deemed unreliable. Focusing just on the first 6 revolutions, however, Figure 14 shows how the flow field changes for an axial slice after the 3rd rotor.

Figure 15: Rapid startup URANS calculation after 6 rotor revolutions

Beginning with the fifth revolution, the four stall cells become three stall cells. This process is completed by the 6th revolution as there is one very large stall cell and two smaller ones. Finally, the computed rotating speed of the three stall cells was slower than the normal startup condition with its four stall cells. Although there is no data with three stall cells to compare to directly, the trend of a reduced rotating speed coinciding with a reduced number of cells mirrors the test data.

Thus we can conclude that the CFD has potential to capture the trend that a rapid startup leads to a decrease in the stall cell count. A reduced stall cell count usually means an increase in the pressure fluctuation magnitude and thus an increase in blade vibrations. This implies that URANS calculations in the future could be useful in designing a rapid startup schedule that avoids catastrophic events like surge or blade resonance.

In order to actually determine if one stall cell can actually be predicted by CFD, a longer compressor model will have to be used to ensure reversed flow doesn’t reach the exit boundary or the boundary condition will have to be modified to handle such flows. Research is still ongoing on this topic.

Figure 16: Axial cut after the 3rd Rotor showing the change in stalling behavior moving from the normal startup condition to the rapid startup condition
CONCLUSION

The major conclusions of this research are as follows:

(1) The URANS calculation for the normal startup condition matched the performance data as well as the casing static pressure data implying that the CFD is capturing the overall compressor matching at startup (i.e. at 50% of design speed with the IGV through 3rd stator variable vanes severely closed).

(2) The URANS calculation predicts a stable four cell rotating stall flow pattern after 10 revolutions. The number of stall cells and the rotating speed of the stall cells match the test data implying that CFD is able to capture the unsteady flow behavior as well.

(3) For the rapid startup condition, after 6 rotor revolutions the URANS shows a similar trend as the test data which is a decrease in the number of rotating stall cells compared to the normal startup condition.

In summary, for the industrial gas turbine compressor under study we have shown that a full annulus URANS calculation has the potential to simulate the complex unsteady flow during startup and capture the trends seen in the test data for the rapid startup condition. Research into this topic is still ongoing, but we can conclude that CFD is a powerful tool in studying under what conditions it is safe to start a gas turbine engine.

REFERENCES


